

# NUMERICAL SIMULATION OF LIQUID ATOMISATION

WAN MUHAMMAD MUKHLIS BIN WAN AB LATIF

Report submitted as partial fulfillment of the requirements  
for the award of the degree of  
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering  
UNIVERSITI MALAYSIA PAHANG

DECEMBER 2010

**UNIVERSITI MALAYSIA PAHANG**  
**FACULTY OF MECHANICAL ENGINEERING**

I certify that the thesis entitled “Numerical Simulation Of Liquid Atomisation” is written by Wan Muhammad Mukhlis Bin Wan Ab Latif. I have examined the final copy of this thesis and in my opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. I herewith recommend that it be accepted in fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

DR. KORADA VISWANATHA SHARMA

Examiner

Signature

### **SUPERVISOR'S DECLARATION**

I hereby declare that I have checked this project report and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering

Signature : .....

Name of Supervisor : TN HAJI AMIRRUDDIN ABDUL KADIR

Position : DEPUTY DEAN (ACADEMIC & STUDENT  
DEVELOPMENT AFFAIRS)

Date : 6 DECEMBER 2010

### STUDENT'S DECLARATION

I hereby declare that the work in this thesis is my own except for quotations and summaries which have been duly acknowledged. The thesis has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature : .....  
Name : WAN MUHAMMAD MUKHLIS WAN AB LATIF  
Id Number : MH08004  
Date: : 6 DECEMBER 2010

## ACKNOWLEDGEMENT

First and foremost, I want to thank Allah S.W.T, for giving me the source of power, knowledge and strength to finish the project and dissertation for completing my Bachelor of Mechanical Engineering final year project.

I would like to express my gratitude to my supervisor Hj. Amiruddin Abdul Kadir for his wisdom, endurance and encouragement during his supervision period. He guided me through the dissertation process, never accepting less than my best efforts. His suggestions have helped me improve my technical writing and presentation skills.

Special thank to my lecturers, Mr. Mohd Fadzil Bin Rahim and Mr. Mohd Zuhairi for their kindness in sharing their knowledge about ANSYS Fluent software and doing me a favor in getting additional information for the project. Also thanks to my fellow friends for their co-operation and help.

Lastly, thanks to my family for giving me supports and advice to me to keep looking forward when I'm facing a lot of problems and boundaries in completing my PSM

## ABSTRACT

The focus of this study was to investigate the spray characteristics and atomization performance of gasoline fuel (G100) and ethanol fuel (E100) in a high pressure chamber. The overall spray and atomization characteristics such as an axial spray tip penetration, spray width, and overall SMD were measured experimentally and predicted by using ANSYS Fluent. The development process and the appearance timing of the vortices in the test fuels were very similar. Moreover, the increased injection pressure induced the occurrence of a clear circular shape in the downstream spray and a uniform mixture between the injected spray droplets and ambient air. The axial spray tip penetrations of the test fuels were similar, while the spray width and spray cone angle of G100 were slightly larger than the other fuels. In terms of atomization performance, the E100 fuel among the tested fuels had the largest droplet size because E100 has a high kinematic viscosity and surface tension.

## **ABSTRAK**

Fokus kajian ini adalah untuk mengetahui ciri-ciri semburan dan prestasi pengatoman bahan bakar petrol (G100), dan bahan bakar etanol (E100), dalam ruangan tekanan tinggi. Keseluruhan semburan dan ciri-ciri pengatoman seperti penetrasi hujung paksi semburan, lebar semburan, dan SMD keseluruhan diukur secara eksperimen dan diramal dengan menggunakan ANSYS Fluent. Proses pembangunan dan masa penampilan vortisitas dalam ujian bahan bakar sangat mirip. Penetrasi hujung paksi semburan ujian bahan bakar adalah serupa, sedangkan lebar sembur dan sudut kon G100 sedikit lebih besar dari bahan bakar yang lain. Berkenaan prestasi pengatoman, bahan bakar E100 antara bahan bakar yang diuji mempunyai saiz titisan terbesar kerana E100 memiliki kinematik viskositi dan tegangan permukaan yang tinggi.

## TABLE OF CONTENTS

TITLE	PAGE
<b>TITLE PAGE</b>	i
<b>DECLARATION</b>	ii
<b>DEDICATION</b>	iv
<b>ACKNOWLEDGEMENT</b>	v
<b>ABSTRACT</b>	vi
<b>ABSTRAK</b>	vii
<b>TABLE OF CONTENTS</b>	viii
<b>LIST OF FIGURES</b>	x
 <b>CHAPTER 1:</b>	
<b>INTRODUCTION</b>	<b>1</b>
1.1 Project background	1
1.2 Problem statement	2
1.3 Objectives	2
1.4 Scopes of work	2
1.5 Outline of the project	2
 <b>CHAPTER 2:</b>	
<b>LITERATURE REVIEW</b>	<b>3</b>
2.1 Atomization	3
2.2 Spray parameters	4
2.3 Fuel injection system	5
2.4 Spray simulation	5
2.5 Software simulation	6
2.6 Example simulation	7
2.6.1 Using KIVA-3V	7



2.6.2	Using star-CD	8
2.6.3	Using AVL	9
<b>CHAPTER 3:</b>		
<b>METHODOLOGY</b>		<b>11</b>
3.1	Introduction	11
3.2	Flow chart of methodology	11
3.3	Geometry	13
3.4	Meshing	13
3.5	Setup	14
3.5.1	General	14
3.5.2	Model	16
3.5.3	Material	20
3.5.4	Boundary conditions	21
3.5.5	Solution	23
<b>CHAPTER 4:</b>		
<b>RESULTS AND DISCUSSION</b>		<b>24</b>
4.1	Simulation	24
4.2	Experimental	25
4.3	Comparisons of simulation with experiment result	26
<b>CHAPTER 5:</b>		
<b>CONCLUSION</b>		<b>27</b>
5.1	Conclusion	27
5.2	Recommendation and future work	27
<b>REFERENCES</b>		<b>28</b>
<b>APPENDIX</b>		<b>29</b>

## LIST OF FIGURES

FIGURE NO.	TITLE	PAGE
2.1	Spray regimes	3
2.2	Definitions of spray tip penetration and spray angle	4
2.3	Computational domain	8
2.4	Numerical result	8
2.5	Computational domain	9
2.6	Numerical result	9
2.7	Computational domain	10
2.8	Numerical result	10
3.1	Flow chart	12
3.2	Geometry of domain	13
3.3	Meshing	13
3.4	Mesh display dialog box	14
3.5	Mesh color dialog box	15
3.6	Plane 3 face	15
3.7	Reorder report	16
3.8	Energy dialog box	16
3.9	Viscous model dialog box	17
3.10	Species model dialog box	17
3.11	Discrete phase model dialog box	18
3.12	Tracking tab dialog box	19
3.13	Set injection properties dialog box	19
3.14	Species dialog box	20
3.15	Create/edit materials dialog box	21
3.16	Pressure Outlet dialog box	22
3.17	Wall dialog box	22
3.18	Patch dialog box	23
4.1	G100 (simulation)	24
4.2	E100 (simulation)	25
4.3	G100 (experiment)	25
4.4	comparisons of simulation result with experiment result	26

## **CHAPTER 1**

### **INTRODUCTION**

#### **1.1 PROJECT BACKGROUND**

High pressure spray injection plays a significant role in modern direct injection engines. The detailed understanding of this process becomes even more important in the development of gasoline direct injection engines with a stratified charge concept. Gasoline direct injection engines have attracted considerable attention due to their low fuel consumption and being free of mixture control and meet the strengthening emission regulations. The advantages of the Gasoline direct injection engine were their higher thermal efficiency, better potential for reducing specific fuel consumption, as well as freedom for controlling injection timing and in cylinder fuel quantity. Meanwhile, the Gasoline direct injection engine also has potential for significant improvement of pollutant emissions and start-acceleration performance compared with those of the traditional gasoline engine.

The aim of this project was to illustrate the liquid atomization of the spray influential parameters on the spray characteristics using different ratio gasoline-ethanol blend. The physical properties of gasoline, ethanol and their blend, such as density, viscosity, surface tension, and speed of sound, were measured and used in the numerical simulations. Injection process parameters such as injection pressure, nozzle needle lift, injection rate, and volume of injected fuel were controlled on the fuel injection systems test bench. The simulation results were compared with the experimental result for verification.

## **1.2 PROBLEM STATEMENT**

Sprays have been studied for more than a century but were still under research. Through studies by different researchers, it was found that the spray was influenced by a large number of parameters for example different fuel blend, internal nozzle flow including cavitations, spray velocity profile, turbulence at nozzle exit plus physical and thermodynamic states of liquid and surrounding gas.

## **1.3 OBJECTIVE**

- a) To simulate the spray of gasoline and ethanol fuel.
- b) To compare the simulation result with the experimental result.

## **1.4 SCOPES OF WORK**

This project focused only on high pressure spray. CFD simulation had been conducted in this project using ANSYS Fluent software. Two different type of fuel was used, gasoline (G100) and ethanol (E100). The “E” designates ethanol and the number next to E designates the volume percentage of ethanol. The G100 mean that 100% gasoline and E100 means pure ethanol.

## **1.5 OUTLINE OF THE PROJECT**

In this present chapter the background for investigating high- pressure sprays in gasoline direct injection engines was given. In chapter 2 the theory of spray were presented together with a review of the simulation finding for high pressure sprays. In chapter 3 the CFD code of ANSYS fluent was presented, together with the spray model used, which relates to high pressure sprays. The numerical simulation had been in chapter 3. In chapter 4 was review of experimental and simulation result for high-pressure spray. Chapter 4 also includes a discussion of experimental and simulation result. The final chapter 5 would summarize the main result and conclusion and outline the suggested path for future work. Gantt chart was given in Appendix A.

## CHAPTER 2

### LITERATURE REVIEW

#### 2.1 ATOMIZATION

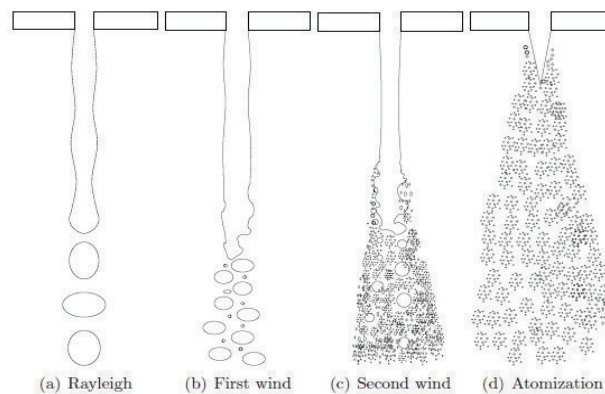
Sprays are usually classified into four spray regimes:

**Rayleigh regime:** Droplet diameter is larger than jet or spray diameter and liquid break up occurs at the downstream of the nozzle.

**First wind induced regime:** Droplet diameter in the order of the spray diameter. Break up occurs at the downstream of the nozzle

**Second wind induced regime:** Droplet diameter smaller is than the spray diameter and break up starts some distance downstream of nozzle.

**Atomization regime:** Droplet diameter much smaller than the spray diameter and break up starts close to the nozzle exit.



**Figure 2.1:** Spray regimes.

Source: Bjarke Skovgard Dam, 2007

Atomization is the process leading to the formation of sprays, which refers to the conversion of bulk liquid into a collection of droplets, often by passing the liquid through a nozzle or an atomizer. Atomization can be considered as a disruption of the consolidating influence of surface tension by the action of internal and external forces. The atomization model supplies the initial conditions for spray computations, in example the drop sizes, velocities, temperatures, and other at the injector nozzle exit.

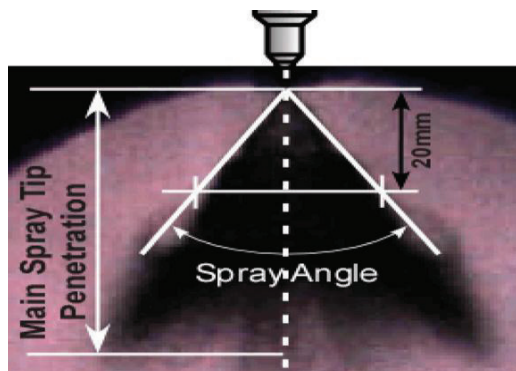
## 2.2 SPRAY PARAMETERS

A number of parameters are defined in order to characterize a spray under certain conditions. Some commonly used parameters are:

**Penetration:** The penetration length is the distance from the nozzle to the end of spray.

**Spray angle:** The spray angle is used to define the size of the spray. It is defined as the quasi steady angle, which is reached after the passing of the spray head.

**Sauter Mean Diameter (SMD):** The droplet size in the spray is usually characterized with its SMD. SMD is proportional to the surface to volume ratio and has the advantage that even if the droplets are not spheres their surface to volume fraction is equivalent to a sphere and therefore they heat up and evaporate in the same way.



**Figure 2.2:** Definitions of spray tip penetration and spray angle.

Source: Jian Gao, et.al , 2006

### 2.3 FUEL INJECTION SYSTEM

The fuel injection system needs to provide different operating modes for the different loads. Fuel injection pressure is very high. This higher pressure values allow a higher penetration and reduce the mean droplet diameter determining a better atomized spray and a good penetration. The high injection pressures will enhance atomization but at the same time produce an over penetrating sprays and wall wetting problems, especially when a sac volume is present. For the unthrottled part-load case, a late injection is needed in order to allow stratified charge combustion, with a well atomized compact spray to control the stratification. A well dispersed spray is desirable, with bigger cone angle and a conical shape. As mentioned before the higher injection pressure is necessary to reduce the Sauter mean radius (SMD) of the liquid spray. To better characterize the spray size distribution the DV90 statistic may also be introduced, which is a quantitative measure of the largest droplets in the spray. It is the droplet diameter corresponding to the 90% volume point, so it gives a measure of the droplet size distribution spread. Gasoline direct injection (GDI) injectors can either be single fluid or air-assisted (two phase) and may be classified by atomization mechanism (sheet, turbulence, pressure, cavitations), by actuation type, nozzle configuration (that can be either swirl, slit, multihole or cavity type), or by spray configuration (hollow cone, solid-cone, fan, multi-plume). [Rossella Rotondi, et.al, 2005]

### 2.4 SPRAY SIMULATION

Sprays have always been a challenge for fluid modelers. Sprays that occur within direct injection engines are typically comprised of a very large number of droplets. Each droplet has unique properties and is subject to complex interactions that are a function of those properties. Due to limited computational resources, it is nearly impossible to take into account each individual droplet in a computational simulation. A variety of strategies has been formulated over the years to address this problem. While details vary from to model, most of these strategies fall into two basic categories: Eulerian-type and lagrangian-type formulation. [Sara dailey bauman, 2001]

The Eulerian-type formulation represents the spray using continuous fields on the same computational grid as is used for the ambient fluid. This formulation is often chosen for its simplicity and ease of implementation. Due to the semi-continuous nature of its formulation, spray properties are typically required to remain uniform, such as isothermal droplets and uniform droplet radii, or to follow other simplifying assumptions. Diverse droplet properties can be taken into account by maintaining multiple fields and transport equations. This type is almost appropriate when concerned about macroscopic behavior of the spray on scales much larger than the average droplet spacing or on scales on the order of the spray penetration length. Averaged information about the spray may be obtained and general spray behavior can be observed. However, the Eulerian approach suffers from numerical diffusion, particularly on coarse grids. [Sara dailey bauman, 2001]

The lagrangian-type formulation is based on a fluid-particle model introduced by Dukowicz. The spray is represented by a collection of computational particles. Each particle in turn represents a parcel of spray droplets that are assumed to have identical properties such as position, velocity, density, radius, and temperature. Often referred to as the discrete droplet model or stochastic particle model, this formulation is more resistant to the numerical diffusion inherent in a semi-continuous field representation. Though the droplets of a single particle have identical properties, each particle in the spray can have a unique set of properties and interact with the fluid accordingly. If appropriately chosen probability distributions are used to define particle properties, an adequate statistical representation of realistic sprays may be obtained when a sufficiently large number of computational particles are used. In the limit of single droplet per particle and assuming appropriate initial conditions are known, this type of formulation approaches the ideal conditions for simulating the spray. [Sara dailey bauman, 2001]

## **2.5 SOFTWARE SIMULATION**

Nowadays computational fluid dynamics (CFD) plays a key role for the optimization of the combustion process in direct injection (DI) diesel engines. Despite their great uncertainties compared to the experimental studies, numerical simulations permit carrying out extensive parametric studies, isolating every single variable involved in the general process at any point in time and at any position in physical space. Modeling also



allows one to artificially separate specific subprocess in example spray atomization, evaporation, diffusive combustion, and emissions from the others that would interact in the real system or to investigate the effects of unnatural boundary conditions on such processes, in order to better understand the combustion process in engines. Basically, engine simulation models can be classified into three categories, depending on their complexity and increasing requirements with respect to the computational power: thermodynamics and phenomenological models, and the multidimensional models used in the so-called CFD codes. [J. M. Desantes, et.al 2009]

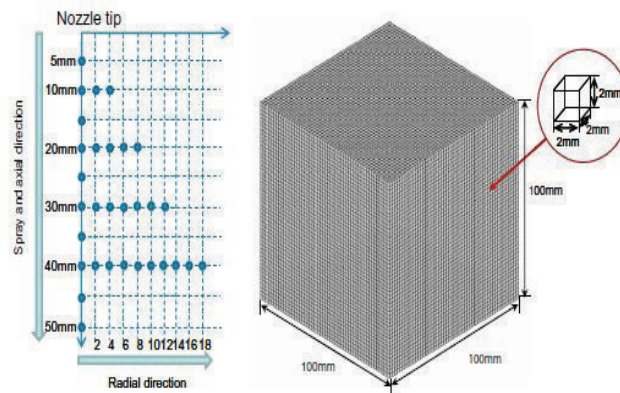
The thermodynamic codes assume that the cylinder charge is uniform in both composition and temperature, at all times during the cycle. These models are computationally very efficient but cannot provide insight into local processes such as the spatial variation in mixture composition and temperature, essential to predict exhaust emissions. Phenomenological spray and combustion models are more complex than the thermodynamic models since they divide the combustion chamber into numerous different zones, characterized by different temperature and compositions. In the multidimensional CFD-codes the full set of differential equations for species, mass, energy, and momentum conservation are solved on a relatively fine numerical mesh with the inclusion of models to account for the effects of turbulence. As a result, these models are best suited to analyze the various subscale processes of mixture formation and combustion with great detail. [J. M. Desantes, et.al, 2009]

## **2.6 EXAMPLE SIMULATION**

### **2.6.1 Using KIVA-3 V**

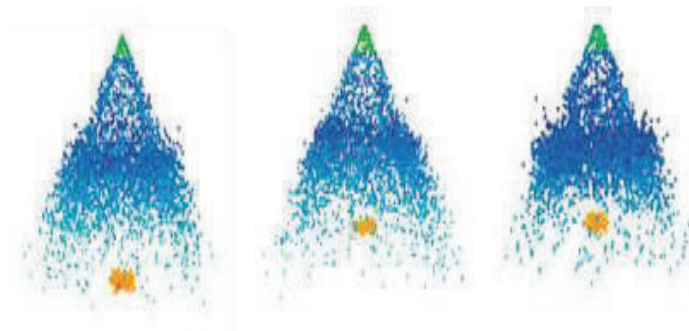
In order to apply the spray of gasoline and ethanol fuels to the KIVA-3 V code, gasoline ( $C_8H_{17}$ ) and ethanol ( $C_2H_5OH$ ) fuel properties from the fuel library (Amsden, 1993) were used to calculate the gasoline (G100) and ethanol (E100) sprays. In the case of the ethanol blended gasoline fuel (E85), the fuel library was created on the basis of measured fuel properties such as surface tension, density, and kinetic viscosity. Therefore, the calculations of the spray characteristics injected through the GDI injector were conducted after the fuel library of the test fuel was added and modified in the KIVA-3V

code. The calculation conditions used were the same as the experimental conditions. The total number of injected droplets was set at 5000. The spray angle and duration of pre-spray with nozzle hole diameter were determined to be 10 degree of a solid cone form and 0.1 ms before the conversion of the hollow cone, respectively. In order to analyze the spray characteristics, uniform cubic meshes with a computational cell size of 2 x 2 x 2 mm were used. [Su Han Park, et.al, 2009]



**Figure 2.3:** Computational domain.

Source: Su Han Park, Hyung Jun Kim, Hyun Kyu Suh, Chang Sik Lee, 2009

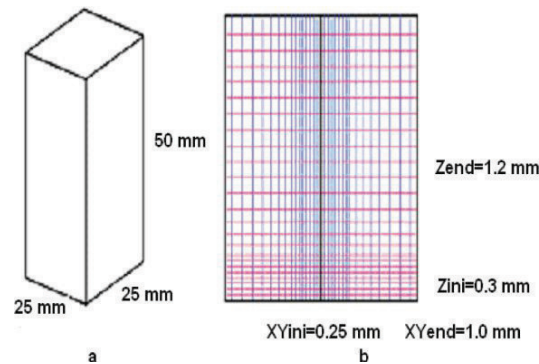


**Figure 2.4:** Numerical result.

Source: Su Han Park, Hyung Jun Kim, Hyun Kyu Suh, Chang Sik Lee, 2009

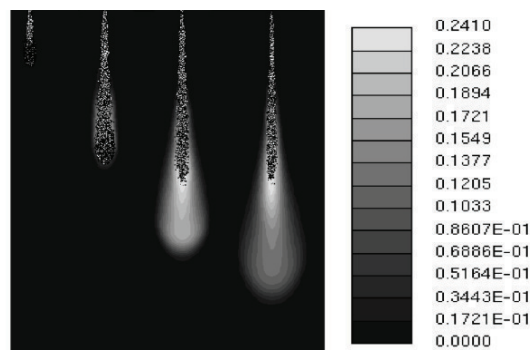
### 2.6.2 Using Star-CD CFD

A cubic computational mesh with variable cell sizes in the  $X$ ,  $Y$ , and  $Z$  directions was generated, with a more refined region at the nozzle exit, similar to that used by different authors to analyze the Lagrangian spray and even gaseous sprays. The mesh is formed by 175 000 cells, and it has a minimum cell size of 0.25 mm at the orifice exit. This mesh is designed for reducing possible errors of droplet positioning in the dense spray region, due to the high relative fuel/air velocity and to the high cell/drop size ratio. With the time step mentioned above and this mesh, a typical spray calculation for an injection event of 2 ms takes about 14 h CPU time with no parallelization. [J. M. Desantes, et.al, 2009]



**Figure 2.5:** Computational domain.

Source: J. M. Desantes, X. Margot, J. M. Pastor, M. Chavez, and A. Pinzello, 2009

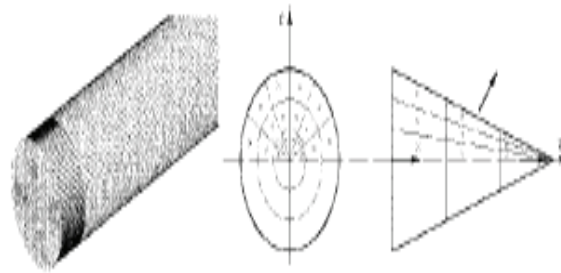


**Figure 2.6:** Numerical result.

Source: J. M. Desantes, X. Margot, J. M. Pastor, M. Chavez, and A. Pinzello, 2009

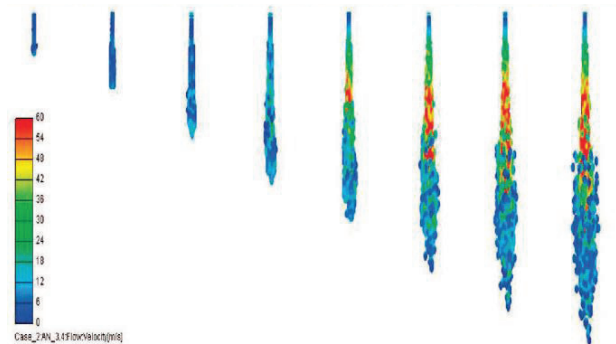
### 2.6.3 Using AVL

The spray was simulated by the AVL 3D using the Euler-Lagrangian approach. With respect to the liquid phase, spray calculations are based on a statistical method referred to as the discrete droplet method. Droplet parcels are introduced in the flow domain with initial conditions of position, size, velocity, temperature, and number of particles in the parcel. The droplets are tracked in the Lagrangian way through the computational grid used for solving the gas phase partial differential equations. Full two-way interaction between the gas and liquid phases is taken into account. A cylindrical mesh with higher density in the middle section and at the nozzle area was used. [Primoz Pogorevc, et.al, 2007]



**Figure 2.7:** Computational domain.

Source: Primoz Pogorevc, Breda Kegl, and Leopold Skerget, 2007



**Figure 2.8:** Numerical result.

Source: Primoz Pogorevc, Breda Kegl, and Leopold Skerget, 2007

## **CHAPTER 3**

### **METHODOLOGY**

#### **3.1 INTRODUCTION**

In this project, several methods were used to complete the project. The methods used were literature survey, data collecting, geometry measurement, conceptual design, computational simulation, and analysis. Each of the methods was explained in details and clearly with explanation.

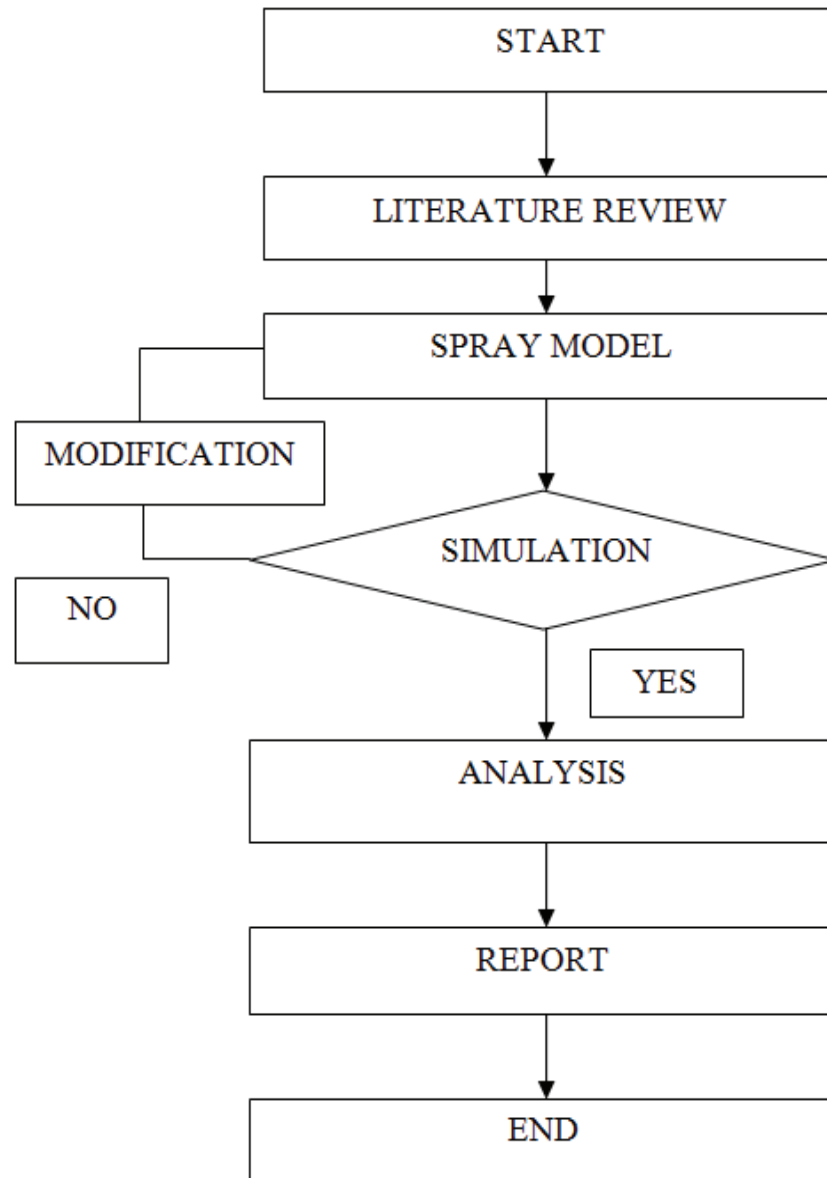
#### **3.2 FLOW CHART OF METHODOLOGY**

The project starts with literature review and research about title such as a review of the concept spray process, fuel properties, injection characteristics, software used, and spray modeling. These tasks have been done through research on the books, journals, technical reports and other sources.

After gathering all relevant information, the project undergoes to spray model. In this step, from the knowledge gathered, the review was used to design the injector, chamber and other to complete the system spray. After completing the spray model, the simulation was run. All results were recorded. When something erred or problems arose in this step, the spray model was modified.

The next step was analysis result. Result from simulation was compared to the experimental result. The comparison of simulation with experimental include the liquid atomization and characteristic of different fuel blend like spray angle, spray penetration, also droplet size.

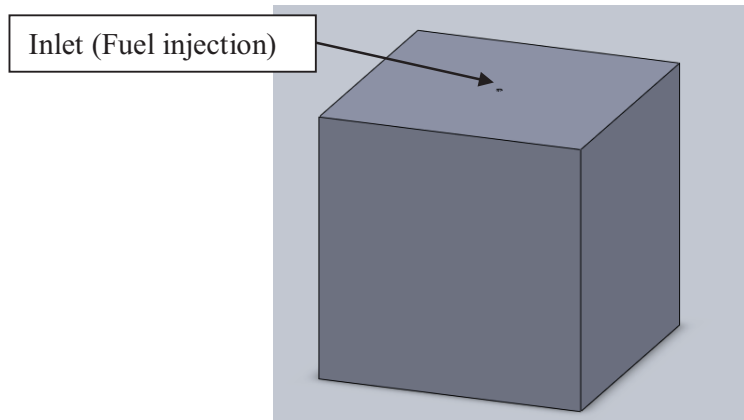
The report was process after complete the analysis. All information like figures, tables and any references were collected to complete the report. Report had been guided by the UMP thesis report writing. This process also included the presentation slide marking for the final presentation of the project. The project ended after the submission of the report.



**Figure 3.1:** Flow chart

### 3.3 GEOMETRY

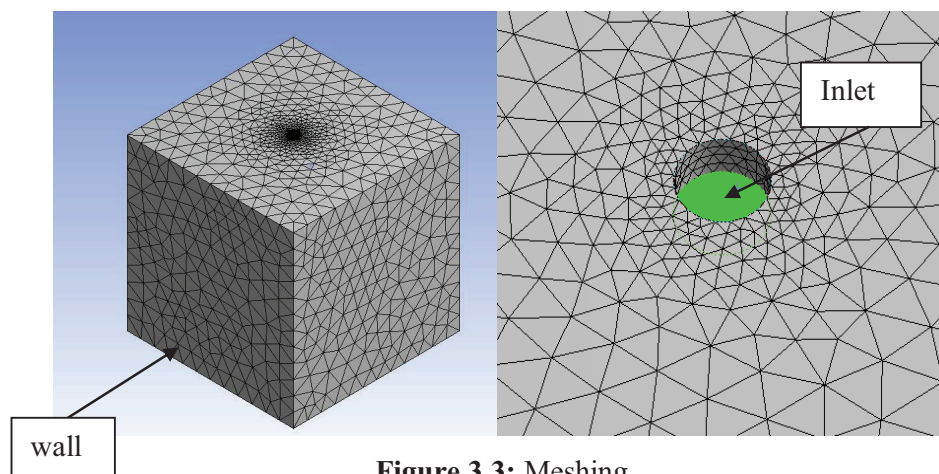
The design of high pressure chamber was produced by Solidwork software. Figure 3.2 show the design of high pressure chamber. The high pressure chamber was designed with cubic shape with 100mm x 100mm x 100mm of dimension. This model was generated to geometry in ANSYS Workbench. The material of model was set to fluid.



**Figure 3.2:** Geometry of domain

### 3.4 MESHING

Model was meshing with automatic mesh in ANSYS ICEM CFD. Consist with two main surface; inlet surface and wall surface. Figure 3.3 show the mesh of model.



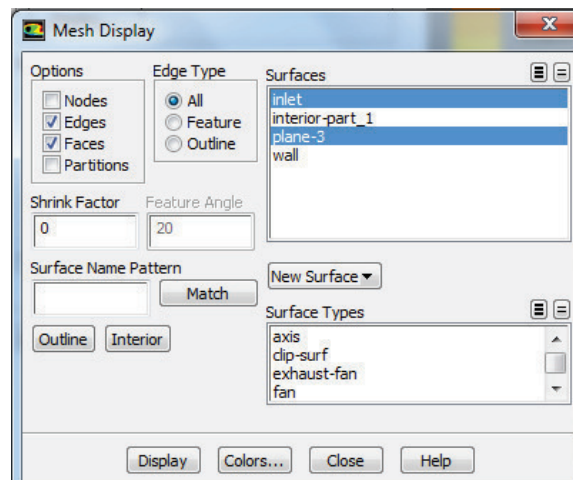
**Figure 3.3:** Meshing

### 3.5 SETUP

After mesh, Setup was used to launch the appropriate application in ANSYS Fluent. All parameters like load, boundary condition, type of material, and otherwise were insert in this setup.

#### 3.5.1 General

- a) Check the mesh. ANSYS Fluent was performed various checks on the mesh and report the progress in the console. Ensure that the reported minimum volume was a positive number.
- b) Display the mesh. Enable Faces in the Options group box. Click display button to open the mesh display dialog box. Select only inlet and plane-3 from the Surfaces selection list.



**Figure 3.4:** Mesh display dialog box

- c) Click the Colors button to open the Mesh Colors dialog box. Select wall from the Types selection list. Select pink from the Colors selection list. Close the Mesh Colors dialog box.